Numerical Simulation of a Coolant Flow and Conjugate Heat Transfer in a Dual-Cooled Fuel Array

Wang-Kee In [a*], Chang-Hwan Shin [a], Tae-Hyun Chun [a]
[a] *Korea Atomic Energy Research Institute*
[*] *Corresponding author: wkin@kaeri.re.kr*

1. Introduction

A research project has started at the Korea Atomic Energy Research Institute to develop an innovative dual-cooled fuel for its employment in an optimized Pressurized Water Reactor (PWR) in Korea, OPR-1000. The dual-cooled fuel for the OPR-1000 is expected to increase the reactor power by 20% as well as to reduce the fuel-pellet temperature by more than 30% without a change of the reactor components other than the fuel.

The thermal-hydraulic challenges for the annular fuel are known to be the splits of the flow and heat between the inner and outer channels [1]. The balances of the flow and heat splits could be optimized by searching for an optimum fuel-rod configuration even though the large sensitivity of the heat split to gap conductances in the inner and outer gaps needs further examination. Another issue associated with the annular fuel is the coolant mixing through the narrow rod-to-rod gaps between the outer channels. This is because the annular fuel rod is larger than a traditional solid fuel in order to provide a sufficient flow rate through the inner cooling channels. Hence, the rod-to-rod gap of the annular fuel decreases for a fixed fuel-assembly size which is compatible with a solid fuel array.

A CFD (Computational Fluid Dynamics) analysis was performed in this study to investigate the effect of a coolant mixing in the fuel rod arrays for two different pitch-to-diameter ratios, i.e., P/D=1.33 (solid fuel) and 1.10 (dual-cooled fuel). The fuel and coolant temperatures were calculated for each of the fuel arrays and compared to each other. This CFD study also predicted the flow distributions in the cooling channels to evaluate the flow velocity in the rod-gap region and the crossflow mixing rate between the adjacent outer channels.

2. Numerical Methods

2.1 CFD Model and Boundary Condition

The solid and dual-cooled fuel arrays for the OPR-1000 are 16x16 and 12x12 rod bundles, respectively. Using a symmetry of the geometry and flow, a 3x3 fuel array with different rod powers was modeled for this CFD study. Fig. 1 shows the plane view of the CFD model for the dual-cooled fuel array with a mixing vane grid. The dual-cooled fuel model consists of annular pellet, dual claddings and inner and outer channels.

A spacer grid with a split vane is considered to investigate the effect of a crossflow mixing downstream of the vane in the outer cooling channel. A CFD model without a spacer grid, i.e., bare rod-bundle model, was also used to evaluate the distributions of the flow and temperature in the outer channel for a fuel array without a flow mixing device. The ratios of the rod-array pitch to the rod diameter (P/D) are 1.33 and 1.10 for the solid and dual-cooled fuels, respectively.

A uniform flow condition and constant pressure were applied at the inlet and outlet boundaries of the fluid regions, respectively. The side boundaries of the outer channel used a cyclic flow condition which allows for a crossflow between the adjacent channels. No slip conditions were used at the fluid and solid interfaces. A constant heat generation rate was imposed at the fuel (pellet) region with the adiabatic external boundaries.

2.2 Numerical Procedure

A CFD code, ANSYS CFX-11.0 [2], was used in this study. ANSYS CFX-11.0 uses a coupled solver, which solves the continuity and momentum equations as a single system in the fluid domain. The energy equations are then solved in both the fluid and solid domains. Turbulence models used in this study are the SSG Reynolds stress model [3] for the bare-rod bundle flow and the SST k-w model [4] for the fuel-bundle flow with the mixing-vane spacer.

Iterative calculations were conducted to obtain converged solutions using a high resolution advection scheme which blends the 1^{st} order upwind and central difference schemes. The iterations were terminated when both RMS residuals of all the governing equations are lower than 10^{-4} and the monitored value of a variable at a specified location is constant.

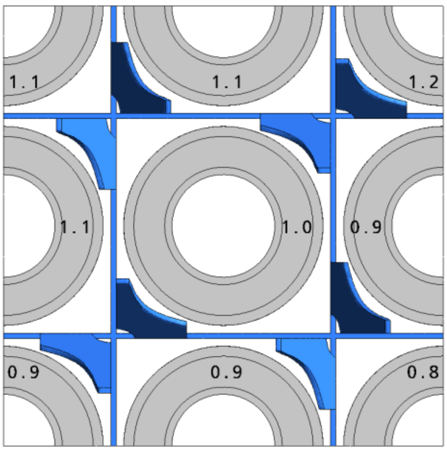

Fig. 1 CFD model for a dual-cooled fuel array with the split-vane grid.

3. Results and Discussion

Fig. 2 shows the fuel temperature and flow mixing pattern for the dual-cooled fuel array 100 mm downstream of the split-vane grid. A strong swirling and crossflow mixing is predicted to occur in the outer channels downstream of the split vane. The swirl pattern inside the subchannel appears to be rather circular for the dual-cooled fuel array (P/D=1.10) while it is elliptic for the solid fuel array (P/D=1.33). This is due to a high flow resistance in the gap region for the tight fuel array with P/D=1.10. The cross-sectional average velocity of the lateral flow is estimated as 16% and 10% of the bulk velocity for the solid and dual cooled fuel arrays. It is also noted in Fig. 2 that the maximum temperature of the hottest fuel pin decreases by more than 500 ^{o}C for the dual-cooled fuel.

Fig. 3 compares the average velocity of crossflow between the adjacent outer channels. The amount of crossflow mixing appears to be the same for both the fuel arrays near the split-vane grid, i.e., Z < 100 mm. However, farther downstream (Z > 100 mm), the crossflow for the dual-cooled fuel array decreases more rapidly than the solid fuel array due to a narrow gap causing a high flow resistance. Approximately 30% of the crossflow appears to decrease for the dual-cooled fuel over the grid span (~400 mm).

Fig. 4 compares the radial distribution of the fuel and coolant temperature through the center-fuel rod. The temperature distributions are symmetric at the center of the each fuel and its variation is large in the fuel-pellet region. This is due to a balance of the thermal resistance in the radial direction and a relatively low thermal conductivity of the pellet. It is also noted that the maximum fuel temperature decreases by more than 450 ^{o}C for the dual-cooled fuel.

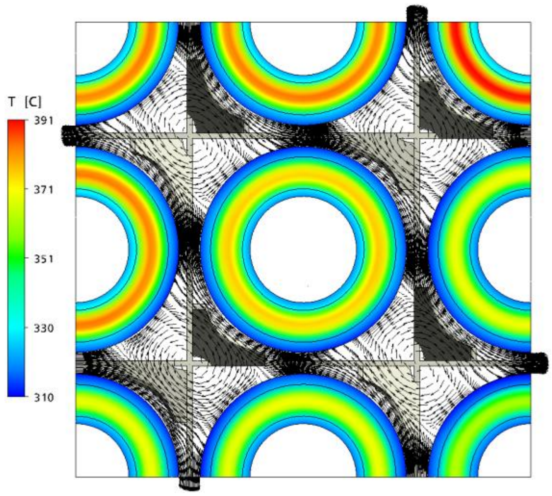

Fig. 2 Flow mixing pattern and fuel temperature for the dual-cooled fuel with the split-vane grid.

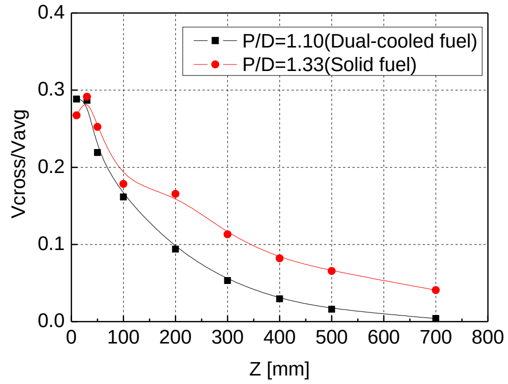

Fig. 3 Average crossflow velocity downstream of the split-vane grid.

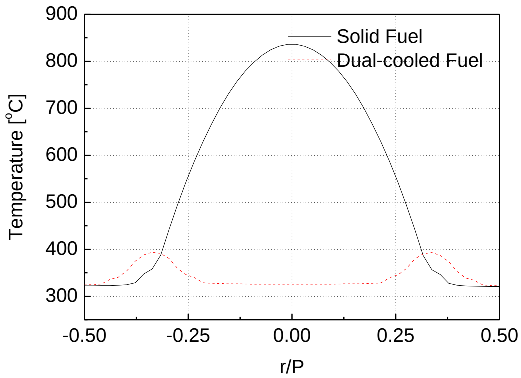

Fig. 4 Radial temperature distribution of central fuel pin.

4. Conclusions

A CFD analysis was performed to investigate the flow and temperature distributions in a dual-cooled PWR fuel array with different fuel-pin powers. Approximately 30% of the crossflow mixing for the fuel array with a mixing vane appears to decrease for the dual-cooled fuel over the grid span (~400 mm). The fuel temperature shows variations of over 600 ^{o}C and less than 100 ^{o}C for the solid and dual-cooled fuels, respectively. The maximum fuel temperature decreases by more than 450 ^{o}C for the dual-cooled fuel.

REFERENCES

[1] D. Feng, P. Hejzlar, and M.S. Kazimi, Thermal-Hydraulic Design of High-Power Density Annular Fuel in PWRs, Nuclear Technology, Vol. 160, p.16, 2007.
[2] ANSYS Inc., ANSYS CFX, Release 11.0, 2006.
[3] C.G. Speziale, S. Sarkar, and T.B. Gatski, Modelling the Pressure-Strain Correlation of Turbulence: An Invariant Dynamical Systems Approach, J. Fluid Mechanics, Vol.227, p.245, 1991.
[4] F.R. Menter, Two-Equation Eddy Viscosity Turbulence Models for Engineering Applications, AIAA J., Vol.32(8), p.1598, 1994.